



SPE 77494

Using Computational Fluid Dynamics (CFD) Simulation to Model Fluid Motion in Process Vessels on Fixed and Floating Platforms

Dr. Ted Frankiewicz (SPE)/NATCO Group and Dr. Chang-Ming Lee/NATCO Group

Copyright 2002, Society of Petroleum Engineers Inc.

This paper was prepared for presentation at the SPE Annual Technical Conference and Exhibition held in San Antonio, Texas, 29 September–2 October 2002.

This paper was selected for presentation by an SPE Program Committee following review of information contained in an abstract submitted by the author(s). Contents of the paper, as presented, have not been reviewed by the Society of Petroleum Engineers and are subject to correction by the author(s). The material, as presented, does not necessarily reflect any position of the Society of Petroleum Engineers, its officers, or members. Papers presented at SPE meetings are subject to publication review by Editorial Committees of the Society of Petroleum Engineers. Electronic reproduction, distribution, or storage of any part of this paper for commercial purposes without the written consent of the Society of Petroleum Engineers is prohibited. Permission to reproduce in print is restricted to an abstract of not more than 300 words; illustrations may not be copied. The abstract must contain conspicuous acknowledgment of where and by whom the paper was presented. Write Librarian, SPE, P.O. Box 833836, Richardson, TX 75083-3836, U.S.A., fax 01-972-952-9435.

Abstract

Computational Fluid Dynamics (CFD) is a mathematical tool capable of simulating a wide range of fluid flows. Integrated CFD software has been applied to study the flows in two- and three-phase oilfield separators. The influence of inlet nozzle configuration, flow distributors, perforated plates, and outlet nozzles have been studied.

Of particular value has been the ability to simulate wave induced sloshing in vessels mounted on floating platforms. The design and placement of baffles to mitigate liquid sloshing was determined by using CFD to simulate fluid flows in vessels destined for service on floating platforms. The simulations accounted for the movement of the vessel based upon its actual location on the platform and identified the influence of process fluid flows on slosh motion. Where possible, laboratory testing has been used to validate CFD simulation results, leading to increased confidence in the mathematical models.

Introduction

Until recently, the engineering design of two- and three-phase separators was considered to be mature. However, the need for separation systems to operate in challenging environments, such as on floating platforms, has led to a demand for improved vessel designs that require reduced fluid residence times and effective separation even when the fluids are jostled by the six-degrees of motion (pitch, roll, heave, surge, yaw and sway) experienced by a floating platform. CFD allows a designer to observe the simulated fluid flow paths within a vessel and assess how the separator will perform under these difficult circumstances.

The application of CFD to solving oilfield related problems

can be of value in new product development, vessel design optimization, determining sources of underperformance for existing or proposed vessels, and for evaluating the projected performance improvement when retrofits are installed to upgrade existing process equipment¹. CFD provides the means to visualize fluid flows within a separator as well as an ability to track the movement of gas/liquid and liquid/liquid interfaces. In newer versions of CFD software, gas bubbles, oil droplets, and solid particles can be tracked through a separator using advanced multiphase models. This permits one to incorporate performance enhancement into the design of a vessel without the need for extensive testing in physical models. Validation of simulated results is recommended when possible, however, to insure that the CFD simulations are in fact providing reasonable predictions.

CFD Simulation Parameters

The simulations reported here were based upon standard “k- ϵ ” turbulence and volume of fluid (VOF) models.² Both steady state and transient simulations were used depending upon the objective of a particular project. The CFD software package used was from Fluent Incorporated (Lebanon, New Hampshire). GAMBIT was used to build the model geometries and their volume meshing. FLUENT was used to run the simulations, and FLUENT's parallel processing capabilities were implemented to accelerate the simulation process.

To model the movement of the vessels on floating platforms, a special User Defined Function (UDF) was developed. The UDF allowed the movement of the separator of interest to be calculated based upon its actual location on the floating platform, the Center of Rotation (COR) of the platform, and the periods/amplitudes for each sea-state induced motion. Typically, the sea-state data for the simulation included 1-year, 10-year and 100-year storm conditions.

Parameters that were derived from the simulation included “drag coefficients”³ at specified wall surfaces within a vessel, average and/or maximum pressure on the surface of an internal component, and fluid velocity profiles. Although these parameters were useful for comparing the intensity of motion on a relative basis, the most instructive information was generally derived from the 2D and 3D animations that showed simulated fluid movement in response to a vessel's

motion. The animations were augmented with particle tracking, a study of fluid velocity vectors and the distribution of turbulence intensity.

Results and Discussion

Key Components. The key components in a typical separator that control fluid path line development include the inlet nozzle, the inlet momentum-breaking device, perforated plates, weir or bucket faces, and outlet nozzles. Some components have been modeled individually as well as in combination during the course of the CFD studies.

CFD Modeling of Perforated Plates. Perforated plates are used in separators both to establish good fluid flow distribution, and to control liquid sloshing. In a CFD simulation, the perforated plate is modeled as porous media of finite thickness with directional permeability. The fraction of open area can be varied in the porous zone, but hole sizes in the plate are not specifically modeled. To validate the assumption that flow through porous media approximates that through a perforated plate, a separate CFD study was conducted on the design and engineering of perforated plates. The use of perforated plates to correct short-circuit fluid flow paths that are generated as a result of fluids hitting a splash plate in a separator has been discussed previously.⁴

Figure 1 shows path lines calculated in a 3-dimensional CFD simulation for fluid flows through a perforated plate in a laboratory test chamber. In this study, physical testing was used to validate the CFD simulation results. The liquid enters the chamber through an off-center inlet. The perforated plate redistributes the flow, but flow path lines develop quickly downstream of the perforated plate and fluid short-circuiting is both predicted by the simulation and observed in laboratory tests. To prevent this short circuiting, a second perforated plate just upstream of the outlet nozzle is required.⁴ A general rule of thumb that has emerged from CFD studies is that the inlet fluid immediately anticipates the outlet and will develop high velocity flow channels unless means to prevent this are installed in the vessel.

Type of Platform Affects Fluid Motion. The type of floating platform on which a separator is installed has a significant impact on the sloshing of fluid inside the separator^{5, 6}. Figures 2A and 2B show representative locations for a large separator on a TLP and on an FPSO as well as the Center of Rotation (COR) for each platform. The length of the moment arm, i.e., the distance from the platform's COR to the separator, affects the intensity of sloshing in a vessel. The moment arm on an FPSO or TLP may be 50 to 200 ft. But the COR of a spar is generally quite low in the water. Thus the moment arm between the COR and the separator can be quite long, 300 to 400 feet not being unusual. Thus, although the degree of pitch for a spar may be low under storm conditions, the long moment arm can result in large periodic spatial displacements of the separator. In one study, a 2-degree pitch for a spar during a 10-year storm resulted in 11 feet of translation for a vertical separator with a period of 10 seconds. This kind of

motion can generate significant momentum waves within a separator that require additional modifications to the vessel internal designs in order to adequately dampen the slosh waves.

A separator installed on an FPSO will generally experience more roll motion than that on a spar or a TLP. The roll motion, which may or may not remain in phase with the pitch and surge, requires the use of special weir designs to control water spillover without interfering with the normal flow of oil over the weir.

One type of slosh motion control baffle that has been installed in separators on floating platforms is a horizontal ring baffle. According to published research^{7, 8}, the ring baffle is capable of dampening liquid slosh motion when properly sized and positioned. CFD simulations confirm a minor reduction in slosh amplitude in the presence of a ring baffle, but the reduction is far less than what is required in a typical 3-phase separator. Other studies⁹ have shown that these horizontal ring baffles can, if improperly placed, generate significant interface turbulence within a vessel that is counter productive for oil-water separation.

CFD Separator Models Including Vessel Internals. Criteria that must be considered for the design and installation of perforated plates include the fraction of opening area of the plate, the size and layout of holes that provide the opening area, the amount of open area under the plate to allow for sand migration, the locations for baffle placement, and the number of baffles required to control flow distribution and/or liquid sloshing.

A separate CFD study was conducted within NATCO to determine a preferable range for the fraction of open area required in a perforated baffle to control fluid flow patterns. Where possible, the results of these studies were compared with standard engineering calculations¹⁰ in order to calibrate and validate the CFD simulations. The targeted fractional open area is a compromise between the need to restrict flow through the plate and the need to minimize the shearing of oil or water droplets to avoid creating emulsions.

The complexity of flow through a perforated plate is illustrated in Figure 3. Figure 3A shows the development and decay of fluid jets through holes in a perforated plate. Note that when flow approaches the plate with a strongly non-uniform velocity distribution, the flow downstream of the plate tends to spread. Figure 3B is a plot of velocity distribution for the liquid as it approaches the perforated plate and then flows through. Each point represents the velocity for the liquid at a specific volume element within the vessel. The top of the plot is essentially the values of maximum fluid velocity at various distances downstream of the plate.

Figure 4 illustrates the high velocity for liquids that can occur under a perforated plate when open area is retained to allow for sand migration. CFD simulation of vessels on floating platforms indicated such flow paths not only favored short-circuiting for water, but also increase the chance of oil/water mixing if the interface level is too close to the lower opening area.

The placement of perforated plates within a vessel is determined by the location and configuration of the inlet nozzle, the placement of the oil weir and water outlet nozzle, the geometry of the vessel, and the natural period of fluid motion within a vessel compared to the actual movement of the vessel due to the movement of the floating platform. Generally, fully symmetric placement of the internal baffles is not desirable.

Figure 5 shows the baffle placements within one particular 10 feet diameter by 40 feet (Seam-to-Seam) three-phase separator. Fluid enters the vessel through a Porta-Test Revolution® Inlet Device. Gas-liquid separation takes place within the inlet device and liquid exits through a horizontal circular slot at the lower end of the capped tube^{1, 4}. The device's vortex tubes are installed in pairs with 6 tubes having been selected for installation in the subject separator.

For the CFD simulation of the separator, the Porta-Test Revolution® inlet devices are simplified as two rectangular blocks with flow entering the vessel from the bottom of the blocks. The first perforated plate is installed just downstream of the Revolution® Tubes in order to redistribute fluids that emerge from the liquid exit slots. The remaining perforated plates are then positioned for additional control of fluid sloshing.

The variation of fluid level mid-way between the properly located perforated plates is modest when the vessel moves in response to the sea-state induced platform movement. Thus the locations of liquid level control and interface level control can be selected based upon an observation of liquid sloshing within a vessel as simulated by the CFD study.

Figures 6A and 6B show how fluid sloshing is predicted in this separator in response to platform movement during a 10-year winter storm condition. The simulation was performed for the vessel with and without perforated plate baffles. Note that without the plates, the fluid motion within the separator borders on violent (this is more evident when watching the animated sequence of fluid motion that was generated from post processing of the CFD simulation results). However, with slosh suppression plates installed, the liquid motion and interface variation within the separator are damped considerably.

As a relative indicator of the intensity of fluid slosh movement, the relative force experienced by the head wall surface inside the vessel can be plotted. Figure 7 shows a superimposed plot of several "drag coefficient" curves on the vessel wall close to the oil-outlet varying with time for different baffle configurations. Note that the drag on the vessel's head wall during a 10-year storm sea state is reduced to about 38% of the drag value where no baffles are installed.

CFD Modeling of Fluid Distributors. The uniform introduction of fluids into a vessel is a critical step in using the entire volume of a separator for its intended purpose. Distributors are often used for this purpose, especially in process components such as electrostatic treaters and filters.

One common emulsion distributor design for electrostatic treaters is to introduce the emulsion uniformly down the length of the vessel using an inverted U-shaped channel referred to as a "box" distributor. Emulsion enters the box from one end, free water falls out of the bottom of the box, and emulsion is distributed into the treater via a series of holes on each side of the box.

Using CFD simulations, followed by laboratory validation tests, it was shown that standard box distributors tend to bypass a significant fraction of emulsion around the electrostatic grid. The CFD simulation results and the confirming laboratory tests are illustrated in Figures 8. Note in each case, the left hand side of the figure shows the path lines that develop as a consequence of using a box distributor configuration.

To improve fluid flow through the electrostatic grid, a shrouded pipe distributor was developed¹¹. The improvement in flow distribution through the grid is evident both in the CFD simulation and in the bench-top validation testing illustrated in Figure 8. This example demonstrates the application of CFD simulations for detecting and correcting fluid flow problems in a separator. Subsequent to these studies, an existing treater in the North Sea was retrofitted with composite plate electrodes and the shrouded pipe distributors. The capacity of the treater was increased by 67% as a result of the installation of these upgrades with a large fraction of this result coming from the improved flow distribution.

Nozzle Design Affects Discharge Liquid Quality. Finally, the design and location of outlet nozzles also affects fluid path lines in a vessel and, as a consequence, the quality of the discharged liquid. Figure 9 shows how fluid approaches a water outlet nozzle in an oil-over-weir separator. Note that water flowing toward the nozzle comes at least in part from near the oil-water interface with the down-coning actually causing a depression of the oil-water interface in the vicinity of the water outlet. This down-coning drags partially separated oil droplets from near the oil-water interface into the discharge liquid, resulting in a degradation of the quality of the water exiting the vessel.

SUMMARY & CONCLUSIONS

CFD is a powerful mathematical tool for simulating the flow within oilfield processing equipment. CFD simulations allow a designer to visualize how all components of a separator, from the inlet nozzle to the outlet nozzle, affect separation and flows. Using CFD, designs for internals such as perforated plates can be developed and their locations within a vessel can be optimized to control flow distribution and to also minimize liquid sloshing for offshore applications.

REFERENCES

1. Frankiewicz, T. C., Browne, M. M., and Lee, C-M., "Reducing Separation Train Sizes and Increasing Capacity by Application of Emerging Technologies", Offshore Technology Conference Paper 13215, 2001.
2. D. L. Youngs, "Time-Dependent Multi-Material Flow with Large Fluid Distortion", K. W. Morton and M. J. Baines, editors, *Numerical Methods for Fluid Dynamics*. Academic Press, 1982.
3. Fluent 6.0 User's Guide, Fluent Inc., Lebanon, NH, November 2001.
4. Lee, C-M., and Frankiewicz, T. C., "Using Computational Fluid Dynamics Simulation to Model Fluid Motion in Oilfield Separators", 9th Annual IBC Conference, London, U. K., June 2002.
5. Hunter, David, NATCO Internal Report #T-1364801, "Effect of Platform Motion on 45' x 11' Separator, North Sea", October 1973.
6. Hunter, David, NATCO Laboratory Test Report #T-95405-05 "Wave Height Determination", April 1974.
7. Silverman, S., and Abramson, H. N., editors, "The Dynamic Behavior of Liquids in Moving Containers, Chapter 2 – Lateral Sloshing in Moving Containers", Report NASA SP-106, Washington D.C., 1966.
8. Silverman, S. and Abramson, H. N., editors, "The Dynamic Behavior of Liquids in Moving Containers, Chapter 4 – Damping of Liquid Motions and Lateral Sloshing", Report NASA SP-106, Washington D.C., 1966.
9. Wallace, H. G., NATCO Laboratory Test Report, "Box-Spar Storage Cell Motion Simulation Study", February 2001.
10. Perry, R. H., Green, D. W., and Maloney, J. O., Editors, *Perry's Handbook Chemical Engineers Handbook*, 6th Edition, McGraw-Hill Book Co., New York, 1984.
11. Sams, G. W., and Wallace, H. G., United States Patent 6,010,634, "System and Method for Separating Mingled Heavier and Lighter Components of a Liquid Stream", January 4, 2000.

ACKNOWLEDGMENTS

The authors would like to thank NATCO Group management in general and Mr. Robert Curcio in particular for supporting this work and for permission to make this presentation. The authors would also like to thank Mr. Gary Sams and Mr. Harry Wallace for their constructive discussions relative to interpreting and validating CFD simulation results and for providing insights that proved valuable in guiding the development of the NATCO's CFD capabilities.

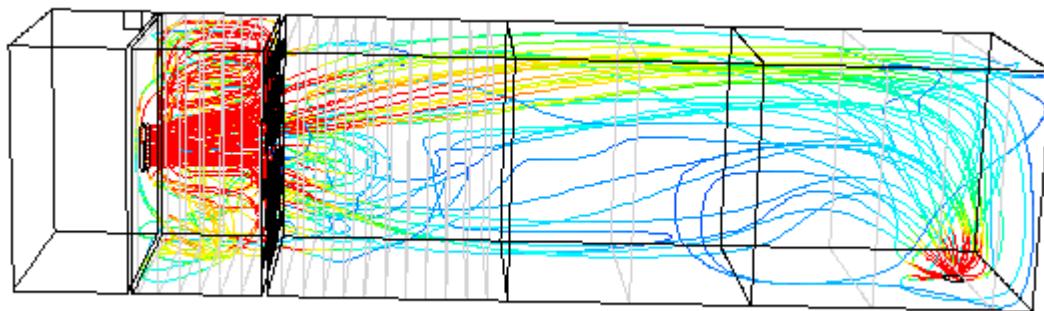


Figure 1. Laboratory model tests validate the CFD calculation of fluid flow path lines, showing how short-circuiting of fluid flow can occur when a single perforated plate is installed in a separator.

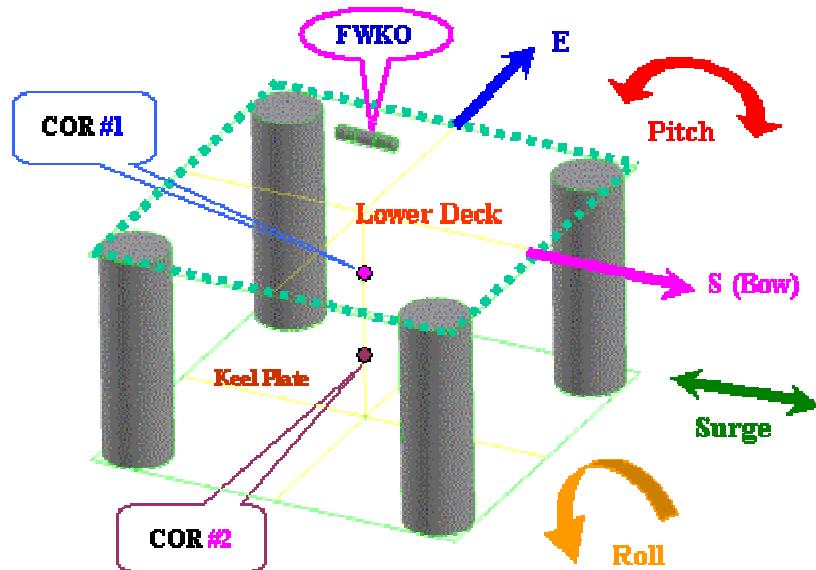


Figure 2A. The placement of a vessel on a TLP is illustrated. Note the position relative to the center of rotation (COR) of the platform.

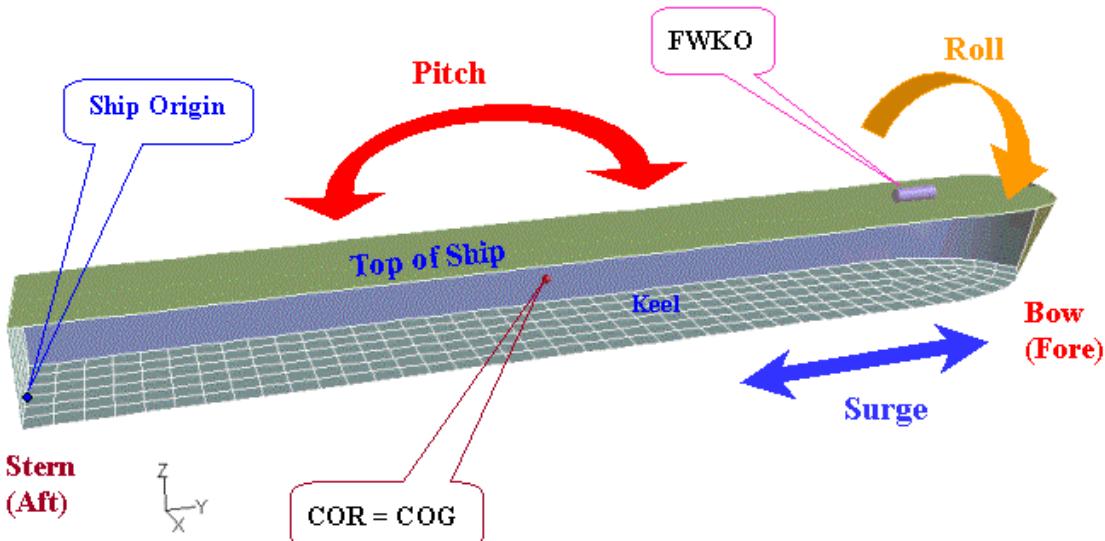


Figure 2B. The placement of a vessel on a FPSO is illustrated. Note the position relative to the center of rotation (COR) of the ship.

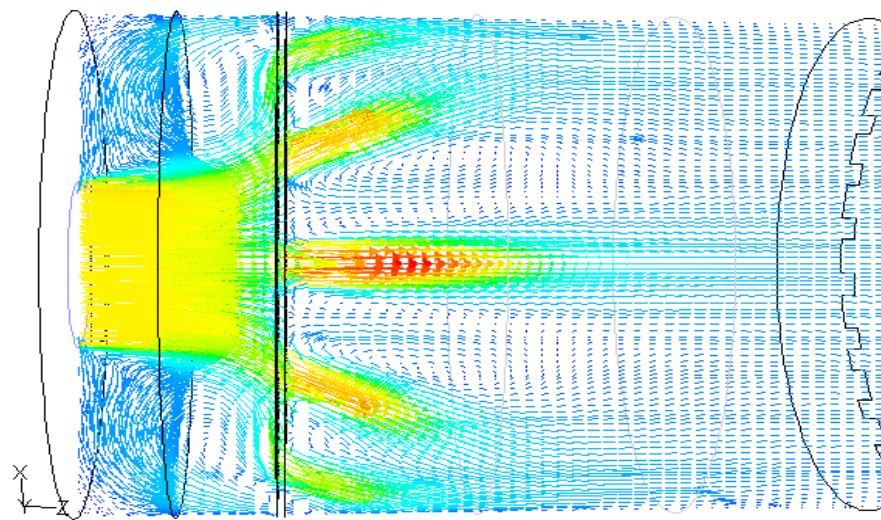


Figure 3A. Fluid flow path lines through a perforated plate are illustrated.

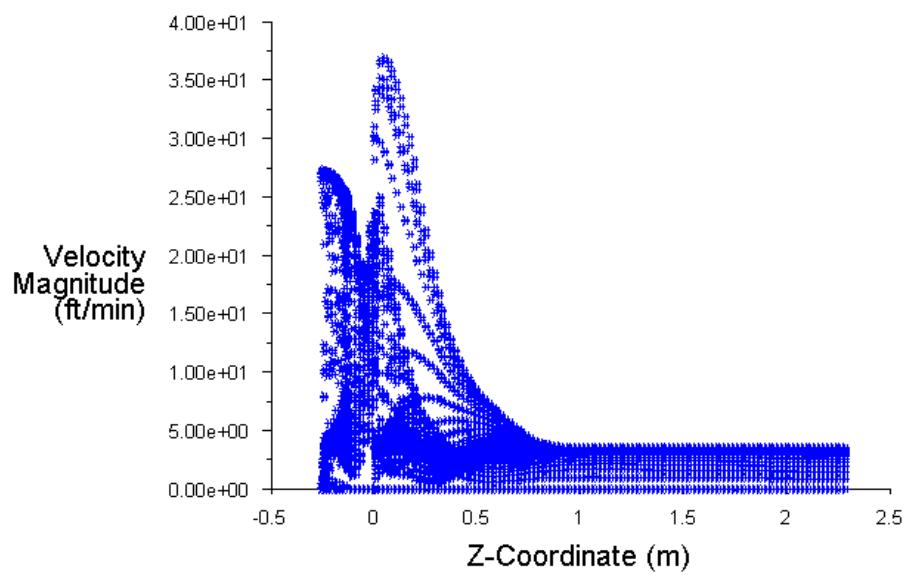


Figure 3B. The velocity distribution for fluid flow through a perforated plate and down the length of vessel is illustrated. Each volume element within the vessel is represented by a data point.

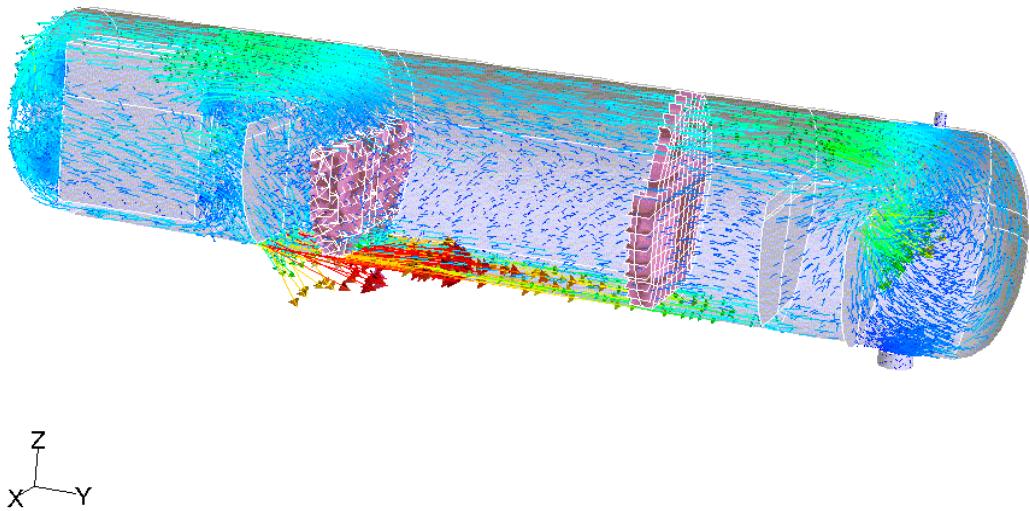


Figure 4. High velocity for water flow can occur under a perforated plate, resulting in reduced water residence time within a vessel.

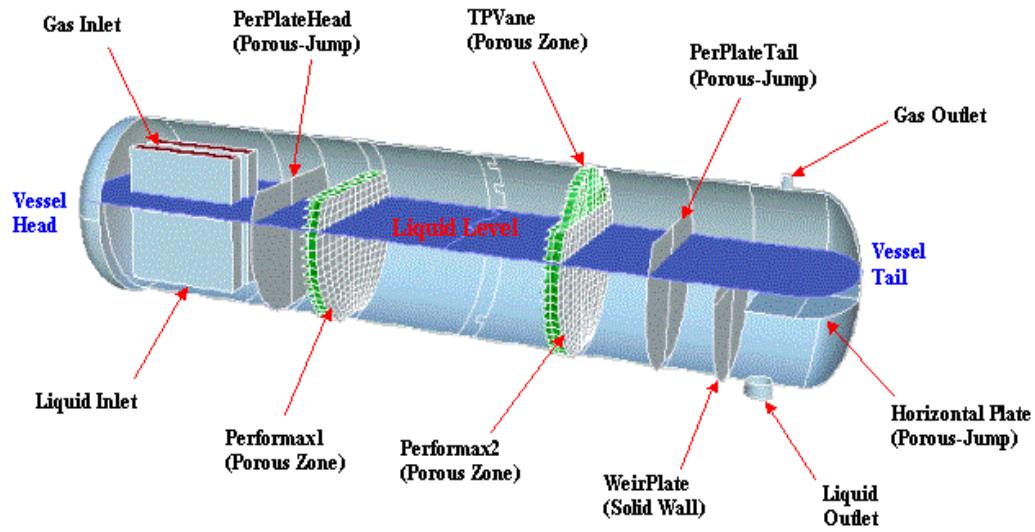


Figure 5. Anti-slosh baffle placements are illustrated for one particular vessel. Baffle location are separator and platform-location specific.

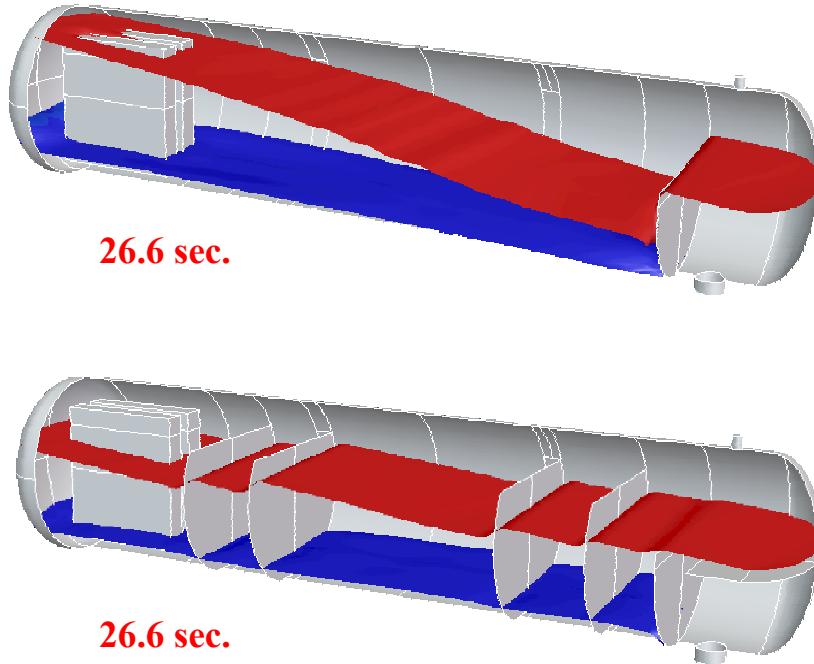


Figure 6. Fluid sloshing in a FWKO on a TLP is shown for the vessel with and without anti-slosh baffling. The drag coefficient on the vessel head was reduced to 38% of its original value by the installation of the baffles, see Figure 9. In both cases, liquid interface positions are shown at simulation time of 26.6 seconds.

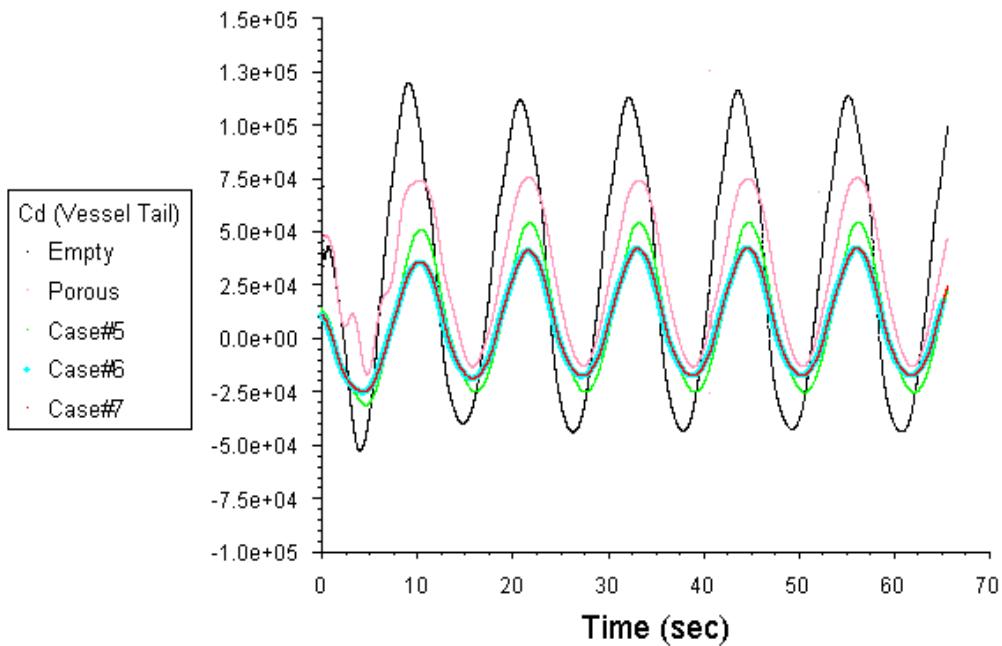


Figure 7. A significant reduction in the drag coefficient on the head of a FWKO can result from the installation of properly engineered and placed anti-slosh baffles.

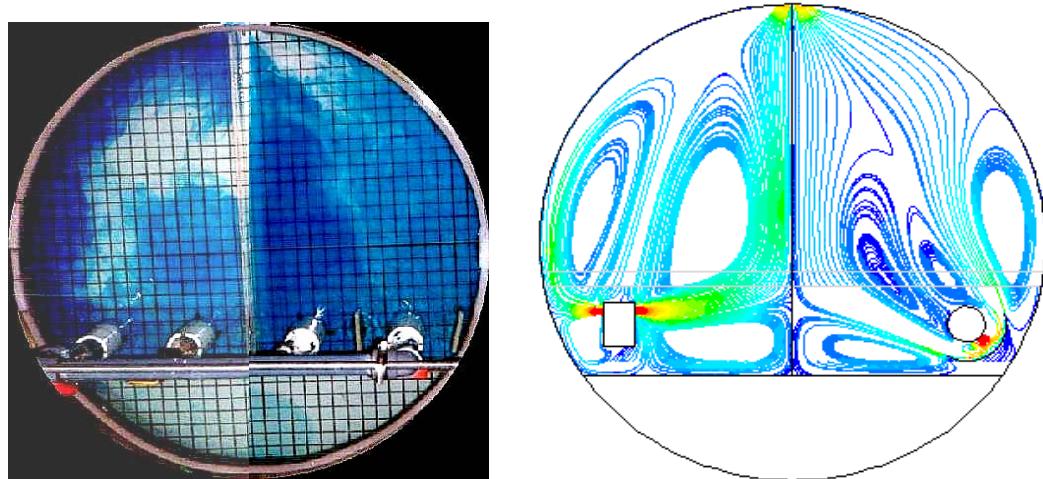


Figure 8. CFD simulation (right) demonstrates how a shrouded pipe distributor improves fluid flow through an electrostatic treater compared to a conventional box distributor. Bench-top testing (left) validated the CFD simulation data. In both cases, the flow distribution from the shrouded pipe is on the right hand side of the illustrations.

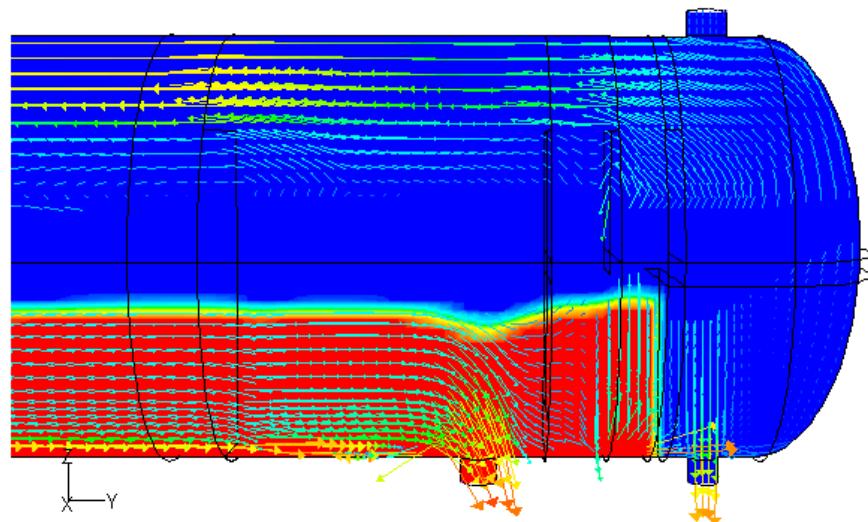


Figure 9. Velocity vectors for fluid approaching a water outlet nozzle illustrate the down-coning of liquid to the nozzle, resulting in reduced discharge water quality.